7. Tool

7.1 Tool functions

General to the tool data

- The tool length in the offsetting record is always taken into consideration in autooperation (according to G17 / G18 / G19 in 3. defined axis).

- In the case of NC program end or NC program abort the spindle tool data becomes theorem into the offsetting record copies.
  Thus tool offset compensations of the NC program (M16) become ineffective.
  When starting of the next NC program is immediately the length of the spindle tool actively (offsetting data).

- The tool length works as shift.
  The indicator position is the programmed position from the NC program.
  The tool length adds itself in the sum shift parameter (P12155).

- The who things radius works as correction.
  The indicator position is smaller or larger than the programmed position from that NC program.

7.1 Tool functions (continued)

T Selection of the group of tools in the NC program

The tool number T is identified and treated with in the analysis. In the tool data array a browsing is started after the suitable group of tools.
When finding the programmed group of tools this record is copied into the PLC tool data array (P8050) and transferred thus to the PLC.

A transfer to the PLC finds however only,

- the first T call in the NC program or
- a modification of the T number.

Special case
If a tool is called, which already is in the spindle (P8100), then the PLC tool data array is filled not from the tool data pool, but the spindle tool record is only copied into the PLC tool record.

Example N10 T5

T extension
For special customer adaptations the T function permits a covered subroutine reference. With each analyzed T, the NC memory is scanned for a Z5. If Z5 exists, then this cycle is covered called. With the call of Z5 the PLC Interface (P8050...) with the new data is described.

7.1 Tool functions (continued)

Tool definitions t...

Existed in the system no tool tools, then can over the identifier, ‘t:’, ‘tr:’, ‘tq:’, ‘tl:’ tool data to be set, with which a radius correction (G41/42) activated afterwards operates. These tool definition can be combined at will in a NC block.
With all functions the tool offsetting record is described (P8150...); i.e. there is temporary tool datas, which are overwritten at a Tx M6 or Tx M1 or at the program end.

For activating everything of these tool tools is not necessary M16.

‘ t: ‘ - Tool radius correction

With the programming of ‘ t: ‘ in the offsetting record only the radius correction of the tool (P8162) is described and activated.
A G41/42 programmed afterwards uses the total of P8160 and P8162 as correction radius!
The quadrant and other tool datas remain unchanged!
This function is practical, in order to correct past tool radius.

Example:

```
N10  ...
N20  t:0.5                          {modification of the past tool radius }
N30  G1 X100 Y100 G42
N40  ...
```

‘ tr: ‘ - Tool radius

With the programming of ‘ tr: ‘ in the offsetting record the radius correction of the tool (P8162) is set to zero and the tool radius (P8160) with the programmed value is described.
The quadrant and other tool data remain unchanged!

Example:

```
N10  ...
N20  tr:50                          { setting the tool radius }
N30  G1 X100 Y100 G42
N40  ...
```

7.1 Tool functions (continued)

Tool definitions t...

‘ tl: ‘ - Tool length

With the programming of ‘ tl: ‘ in the offsetting record the length correction of the tool (P8163) is set to zero and the tool length (P8161) with the programmed value is described.
The quadrant and other tool datas remain unchanged!

Example:

```
N10  ...
N20  tl:80                    { setting the tool length }
N30  G1 X100 Y100
N40  ...
```

‘ tq: ‘ - Tool quadrant

With the programming of ‘ tq: ‘ in the offsetting record the tool quadrant (P8164) is set or modified. Other tool datas remain unchanged!

Example:

```
N10  ...
N20  tr:5   tq:4              {setting a tool radius with quadrant }
N30  G1 X100 Y100 G42
N40        ...
```

7.1 Tool functions (continued)

M06 Tool change

With M06 the tool change is started up. M06 with synchronisation and stop must be defined (see P11051 or P8351).

2 possibilities the tool change consist to execute:

- 1. Tool change in the cycle
- 2. Tool change completely in the PLC implements.

to 1. If the theorem analysis a M06 identifies, it is checked whether the cycle 6 (Z6) exists. There is Z6, then this sub-routine is called, in that the tool change one handles. The actual tool change may not then in the PLC any more with M06 to be executed. But other M function numbers must be used. At the program end of Z6 from the system synchronized and following will become the tool datas taken into consideration (M16).

to 2. If Z6 does not exist, then this means that the PLC with the reception of M06 that Tool change executes. After acknowledgement of M06 the tool datas become taken into consideration (M16).

7.1 Tool functions (continued)

Process of a tool change over PLC (M06 with stop and synchronisation defines):

PLC receives M06

- If a tool is in the spindle, old tools placing.
  This is closed, as P8047 with the tool place is described.

- Subsequently, the tool (T) is changed.
  (tool number and tool place were transferred in the tool data theorem).
  If the change is final, this is acknowledged with the M function-acknowledgement.
  Beforehand P8045 must be however still described.

7.1 Tool functions (continued)

Course of a tool change with Z6

Z6 is called up as sub-routine with M06.

Example: Z6

N10	P8047:1	store tool data
N20	P8045:1	activate tool data
N30	M30	

N10 P8047:1 Store tool data

The tool data theorem in the spindle tool data theorem (P8100) is retransferred into the tool data array. If the spindle tool datas are from T0, one does not retransfer.

N20 P8045:1 Activate tool data

The tool data theorem (P8050...) becomes into the spindle tool data theorem (P8100...) and the offsetting record (P8150...) copies.

N30 M30

With the program end of Z6 the tool data are activated (M16).

7.1 Tool functions

Example for course of a tool change with Z6

NC main program
N...
N...
N100 T3 M6 Call up tool change
N..
N...

When calling up Z6, the parameter area P8050..8099 is actualized with the new tool data of T3.

Z6

N10 _wzwxl_x_pos := 1000; _wzwxl_y_pos:= 300;	Definition of the position for laying down / picking up
N30 P8100=P8050.200	Check if tool-old = tool-new
N40 P8100=0.100	Check if a tool is in the spindle If not: do not store tool data
N50 G00 X:_wzwxl_x_pos Y:_wzwxl_y_pos M01	Approach position for lay down P8120:
N60 P8047:1	Store tool data (Trigger)
N100 P8050=0.200	Check if tool should be picked up. No tool picking up at T0.
N120 G00 X:_wzwxl_x_pos Y:_wzwxl_y_pos M01	Approach position for picking up
N140 P8045:1	Activate tool data

N200 M30

#

7.1 Tool functions (continued)

M16 Tool data call up

M16 with T in the NC block e.g.: N10 T1 M16

During pre-analysis time the corresponding tool data block from the tool data block array is loaded into the actual settlement data block with M16 according to the programmed T-number.
At T0 the settlement data block is cleared.

M16 without T in the NC block e.g.: N10 M16

The settlement data block is activated with M16.

The current spindle tool data block is not changed through M16. M16 can be transmitted to the PLC, if desired. This is not necessary for the function of the tool data call up.
The tool radius is activated with calling up G41 / G42 (tool radius path correciton left / right). The tool - length becomes active at the block end.

e.g.: N100 T2 M16 tool data call up
:
N120 X100 Y100 G42 call up tool radius path correction

7.2 Tool correction

The workpiece programming with tool correction enables the application of tools with different dimensions (example: regrinding of tools).
The dimensions of the tools are indicated with the corresponding T-address. The tool data are calculated by the control on the target path.

This enables the programming of graphically determined workpiece correction and results that the time-costly calculation of base for the cutter center point path is discontinued.

The correction method used for the BWO-control is composed of a combination between intersection - and blending radius correction.

The tool correction is only accomplished in blocks, in which the traverse to be proceeded is unequal to zero (travel differences unequal to zero at straights, radius unequal to zero at circles). The blocks with travels equal to zero are executed at the intersection point or at the end of the inserted transition circle. The amount of one after another lying blocks with travels equal to zero is limited on 12.

Outer contour

Completion of the tool correction
at the beginning of the transition circle

Inner contour

completion of the tool correction at
the intersection point

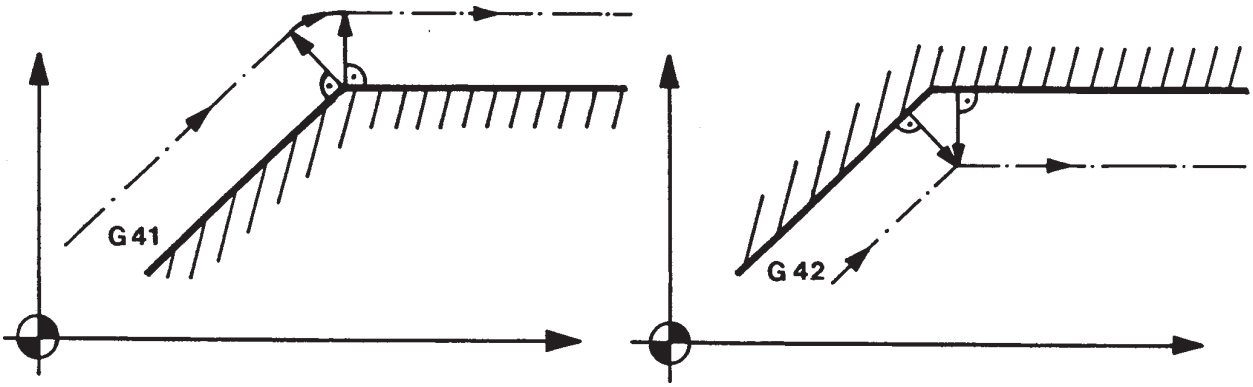


Figure 7-1 Completion of the tool correction

7.2 Tool correction (continued)

Tool dimensions

With M06 / M16 the stored tool dimensions are activated and the tool correction is accomplished.

The tool dimensions of the active tool data are calculated into the position display.
Thereby the position display of the tool axis is the same like the programmed target position, if the axis is driven in position.

The tool axis is determined by the active plane selection of the interpolation main plane. See also G17, G18, G19.

Tool radius

With the traverse conditions G40, G41 and G42 is determined, whether and how the tool radius correction has to be calculated.

Any correction is stopped with G40.
A preceded G41 or G42 is cancelled.

G41 means, that the tool is found on the left side of the programmed path (seen in feed direction).
G42 means, that the tool is found on the right side of the programmed path (seen in feed direction).

The tool radius can be entered positive or negative.
At a positive tool radius the programmed tool correction is calculated.
At a negative tool radius the programmed tool correction is changed: G41 becomes G42 and G42 becomes G41.

The tool radius correction is executed in the indicated interpolation plane. Before changing the interpolation plane correction must be cancelled with G40.

The tool correction can calculate different tool radius, which can be called up with different T-functions.

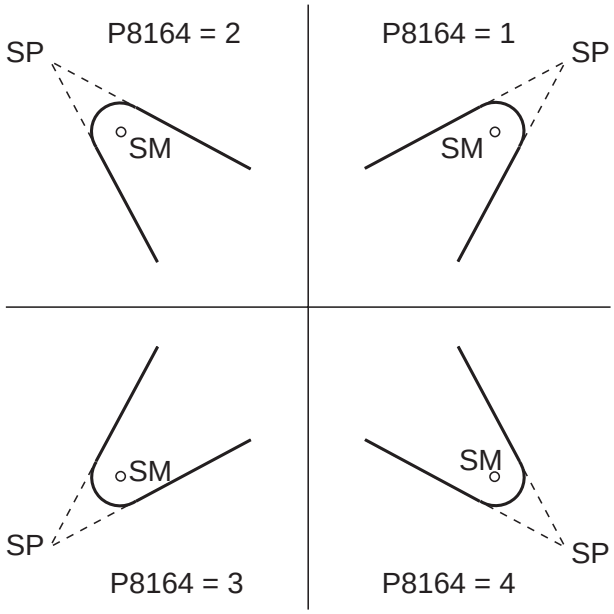
7.2 Tool correction (continued)

Position of the tool

The relative position (quadrant) between tool and workpiece is indicated in P8164.

Definition of the quadrants

P8164 = 1 to 8	quadrant 1 to 8
P8164 = 0 = 9	no quadrant correction SP = SM
SP	theoretical tool peak
SM	radius center point ot the tool



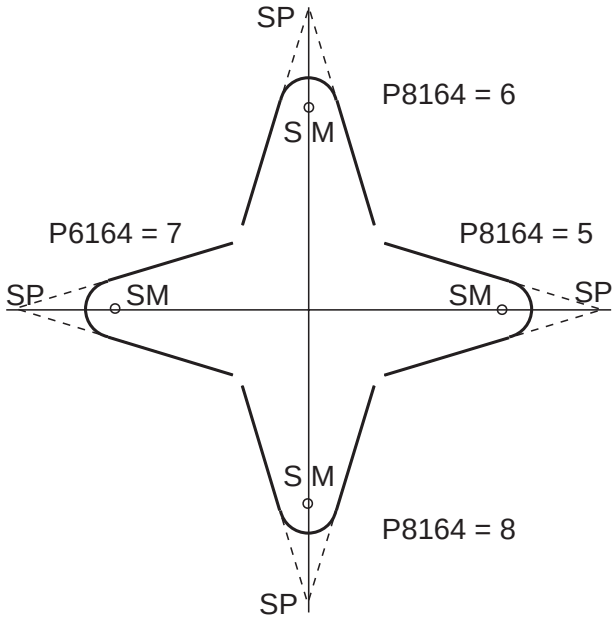


Figure 7-2

7.2 Tool correction (continued)

Special case: Switch off the correction with changing the plane

The switching off of the correction (G40) in a block with travel equal to zero followed by a block with changing plane (G17, G18, G19) leads to wrong positioning of the axes.

Example: N200 G40 Z100	switching off of the correction in plane G17 and travel equal to zero.
G18 X20 Y50	position is approached wrong.
Solution: N200 G40 X40 Y100 Z100	switching off of the correction in plane G17 and travel unequal to zero.
N210 G18 X20 Y50	position is approached right.

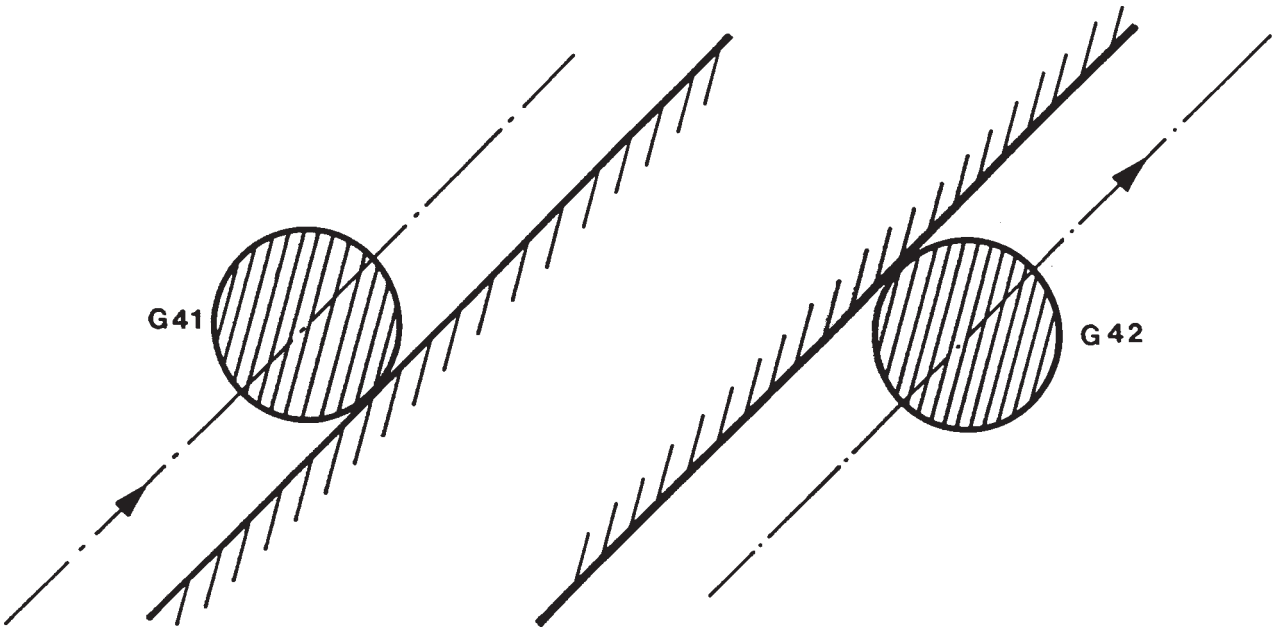


Figure 7-3 Definition of G41 and G42

7.3 Correction principle

If the correction is switched on, the path correction is always accomplished with the same principle:

The corrected path is displaced for the amount of the indicated radius value in reference to the programmed path. The thereby emerging path end points are determined by the cutting of the corrected path elements.

At the outer contour and at changing the correction a transition radius is inserted. At the inner contour the intersection is calculated.

The principle of the correction is explained in the following sketches:

Transition straight - straight

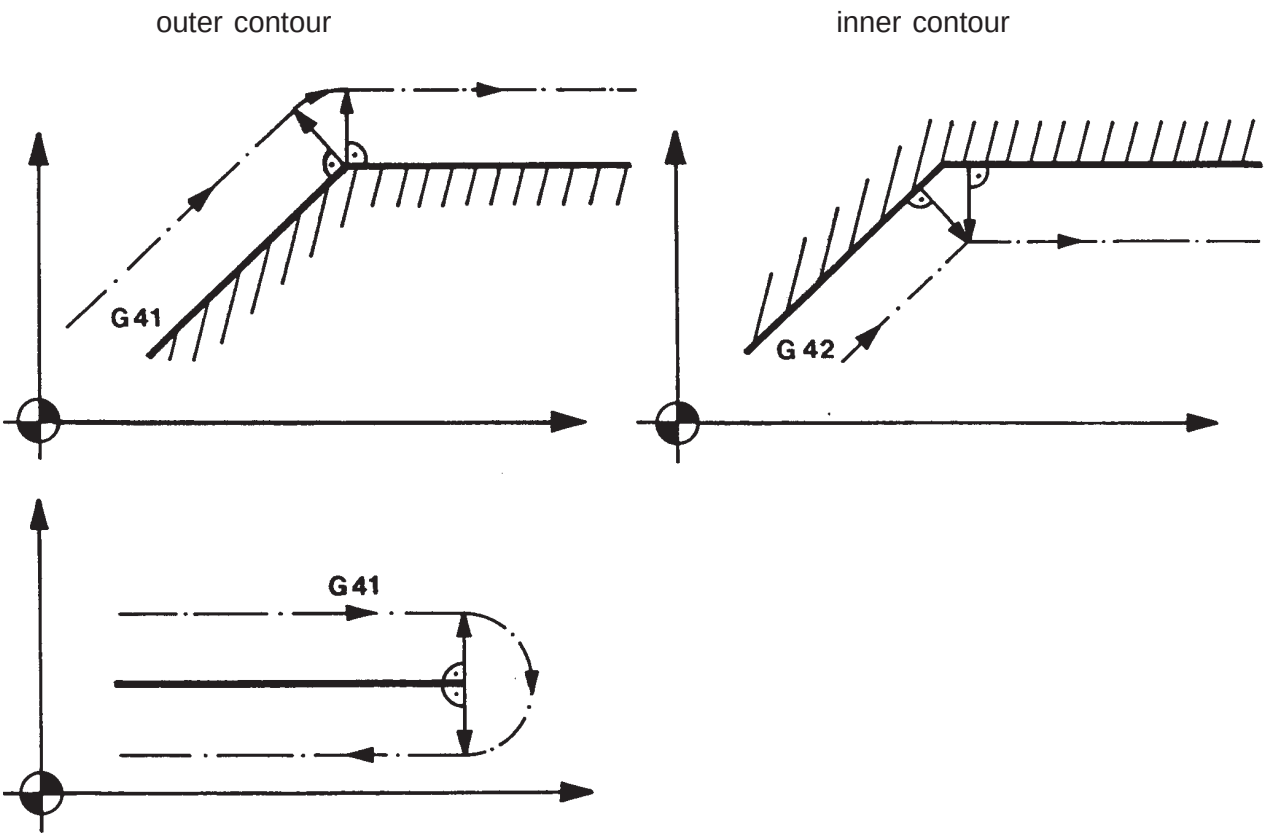
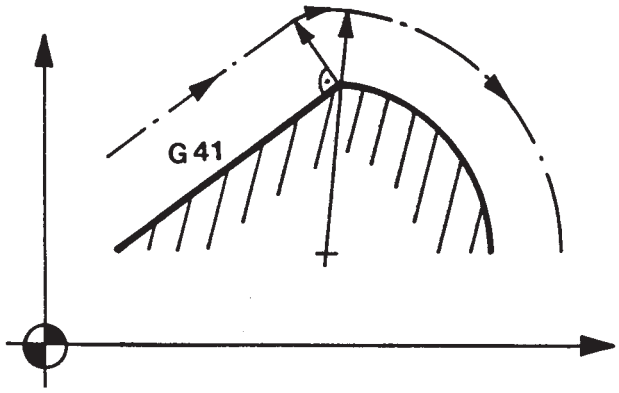


Figure 7-4

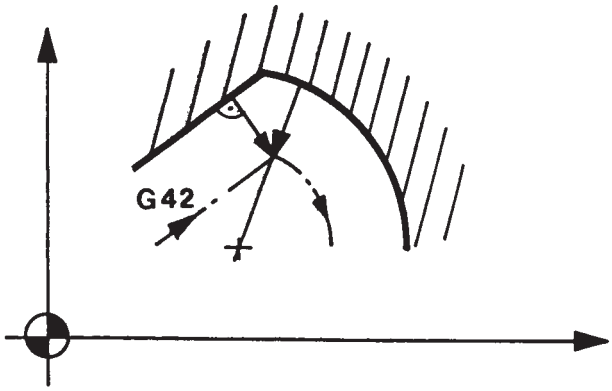
7.3 Correction principle (continued)

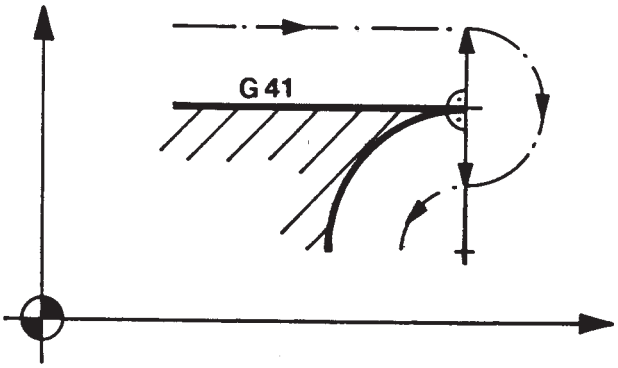
Transition straight - circle (also for circle - straight)

outer contour

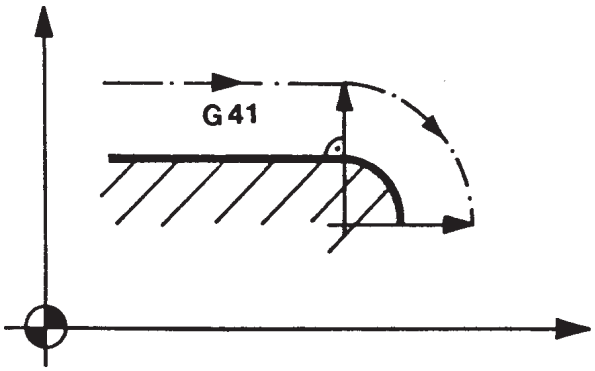


inner contour





Tangential transitions



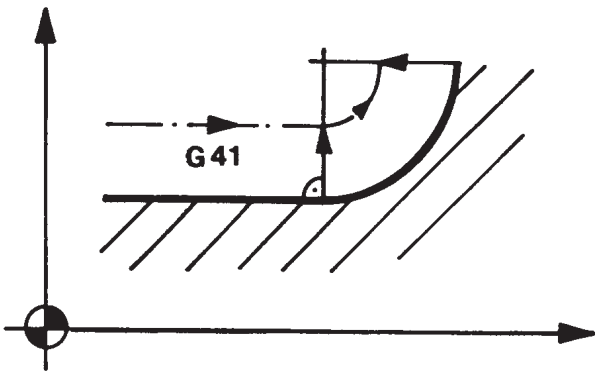
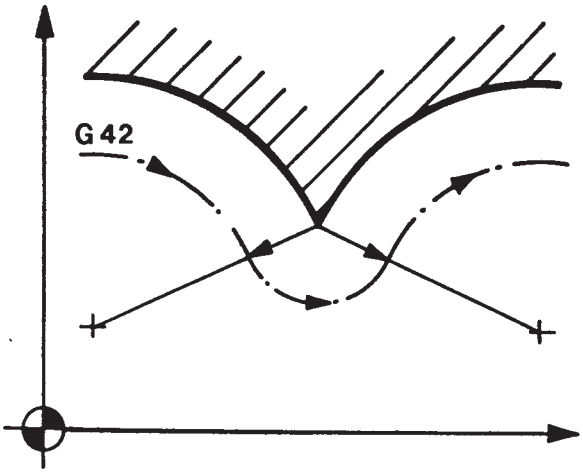


Figure 7-5

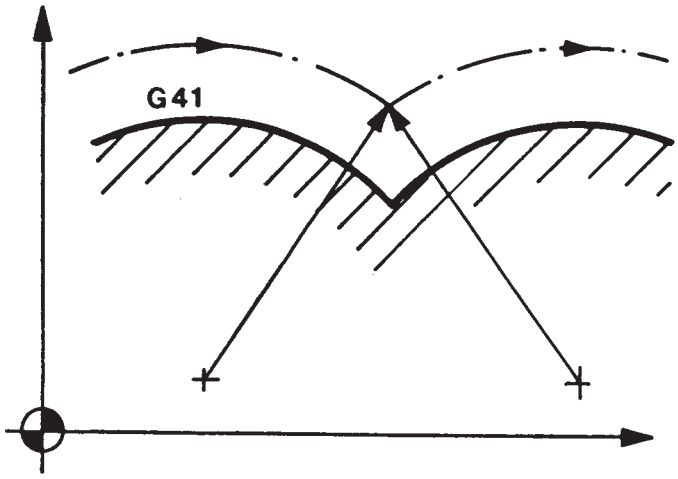
7.3 Correction principle (continued)

Transition circle - circle

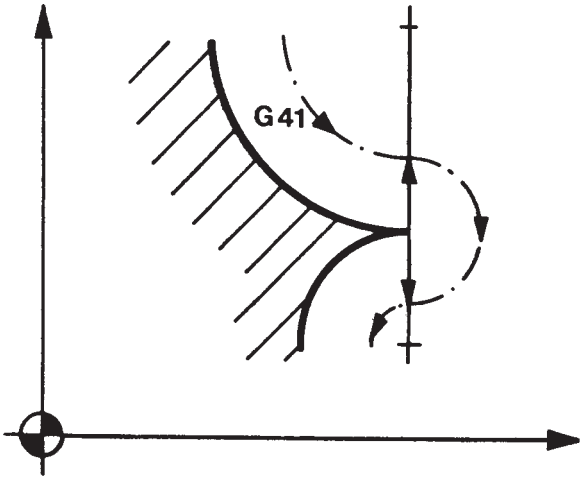
outer contour



inner contour



Tangential transition



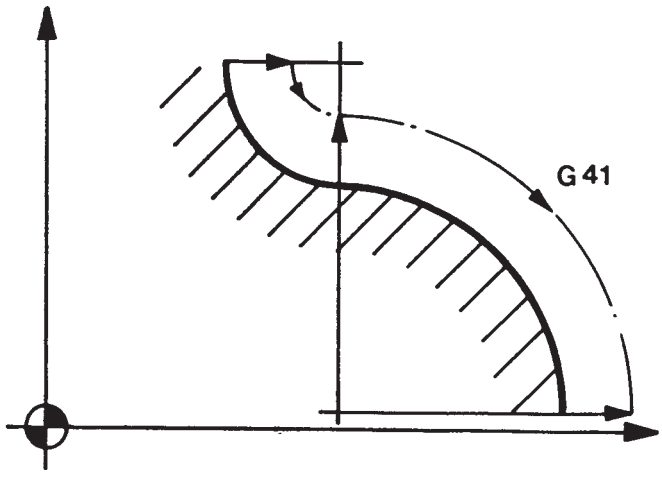


Figure 7-6

7.4 Change of the tool radius

When changing the tool radius, the intersections and transition circles are calculated first with the old radius.
The starting point (old radius) and the end point (new radius) of the corrected path do no more havethe same distance to the programmed path. This is valid for straights and circles.

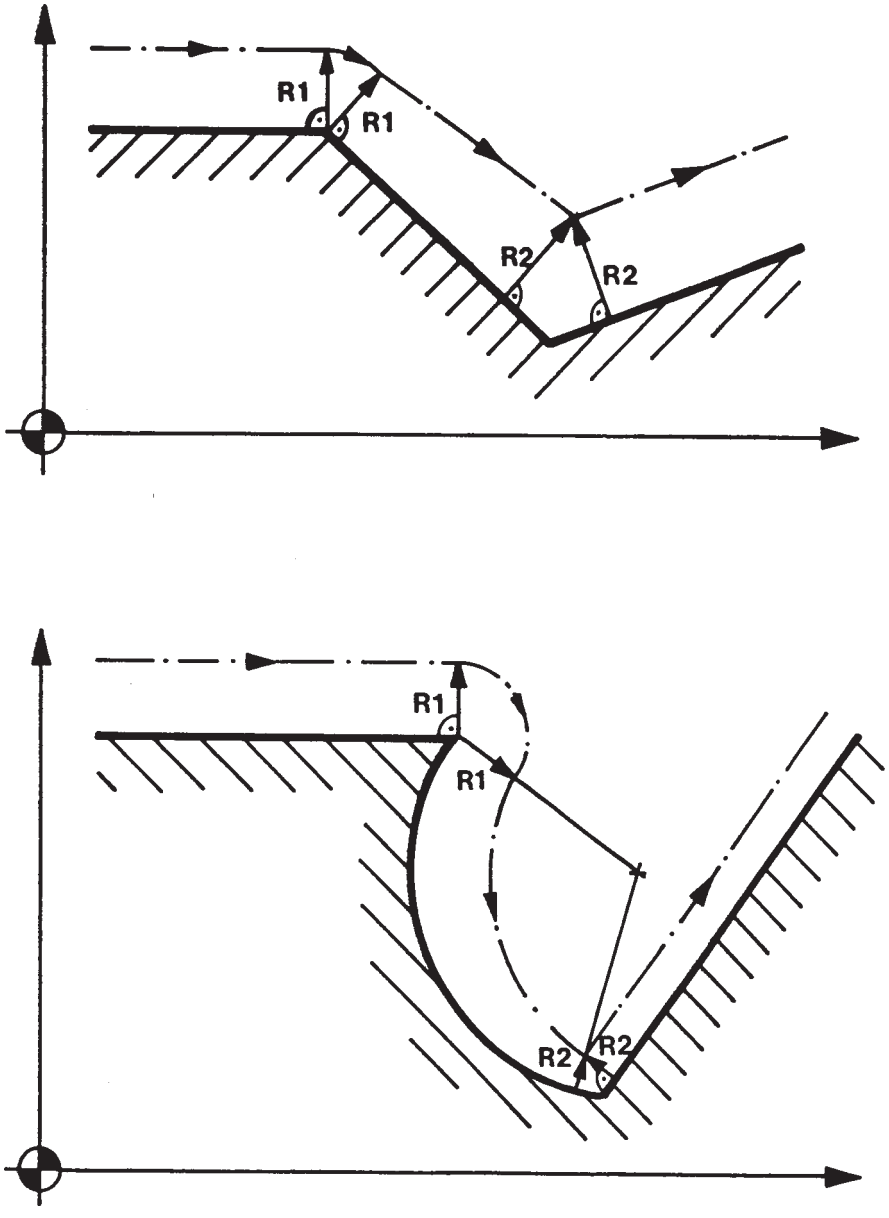


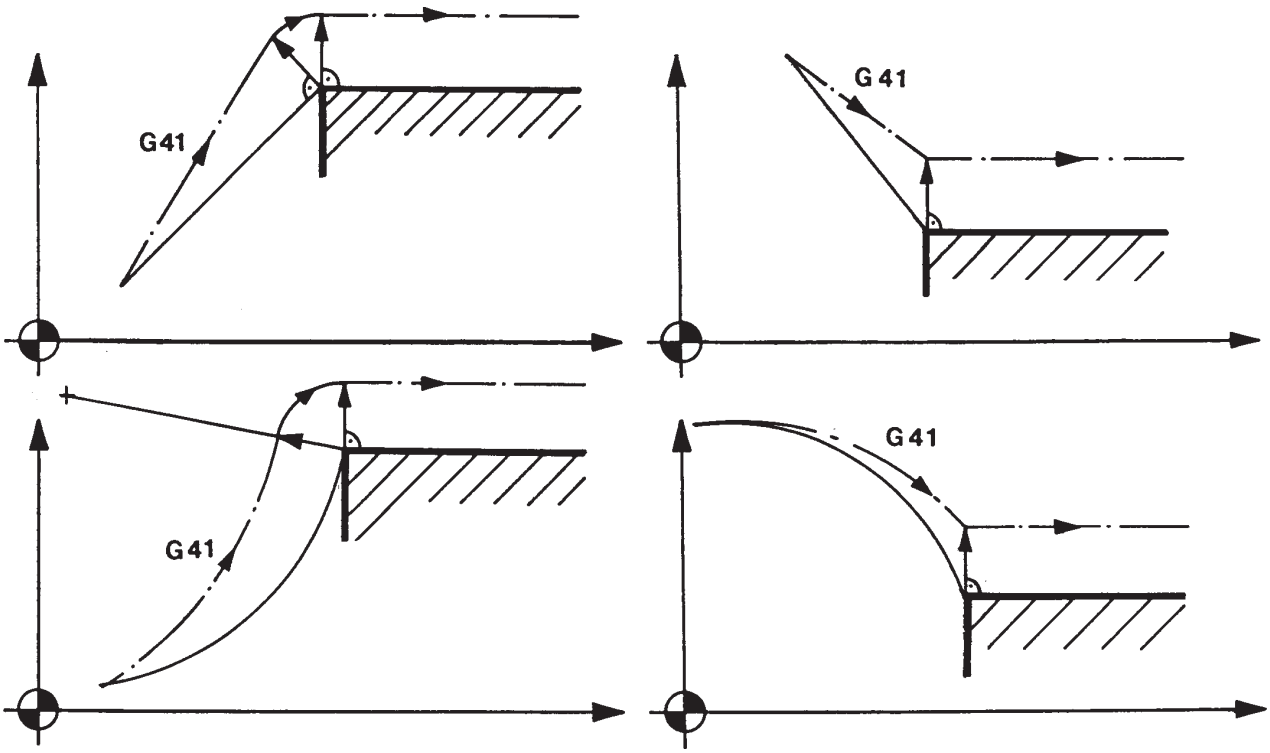
Figure 7-7

7.5 Switching on the correction

The programmed correction is clculated in each block.

outer contour

inner contour



approach tangentially

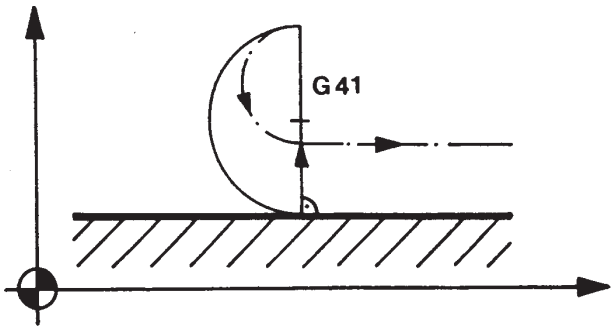
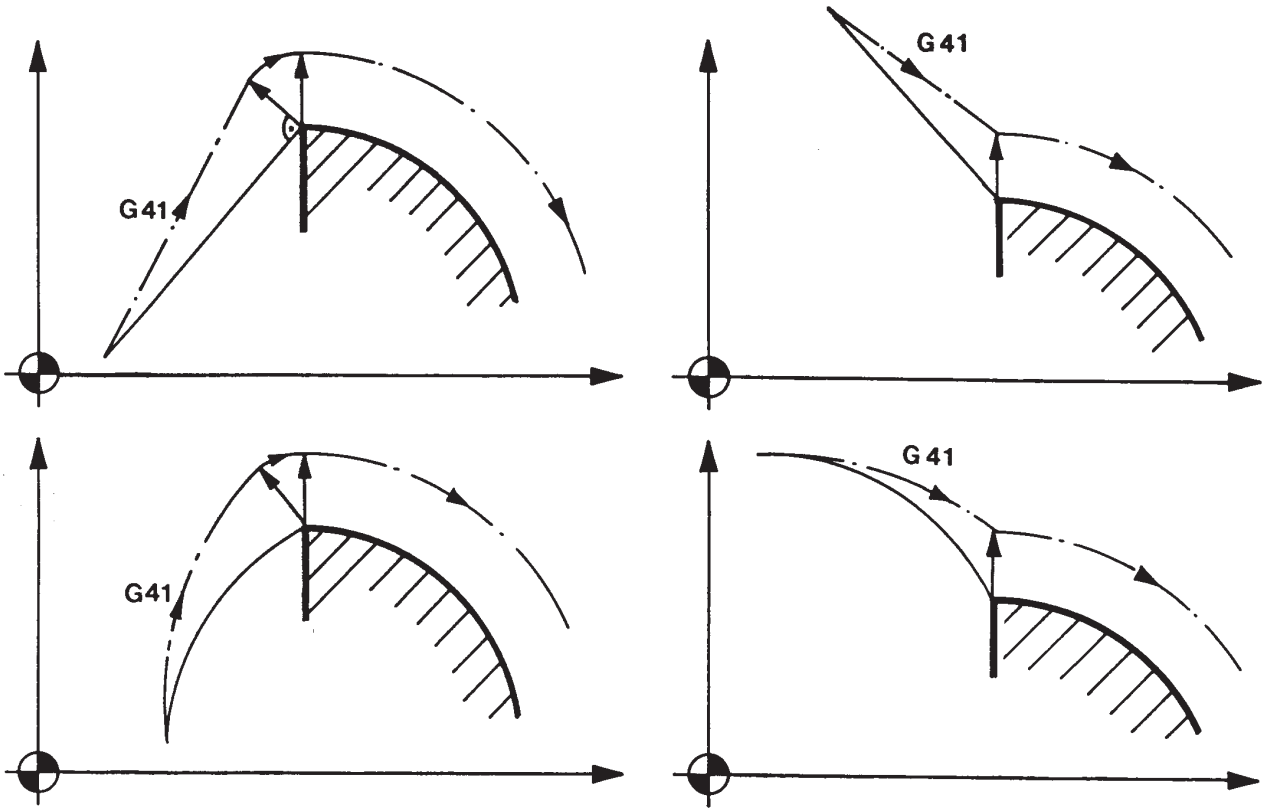


Figure 7-8

7.5 Switching on the correction (continued)

outer contour

inner contour



approach tangentially

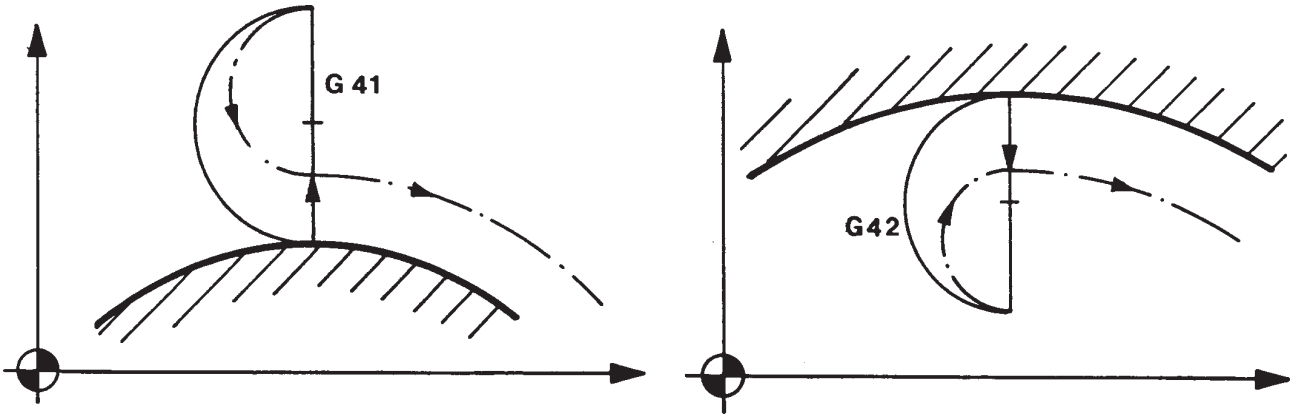
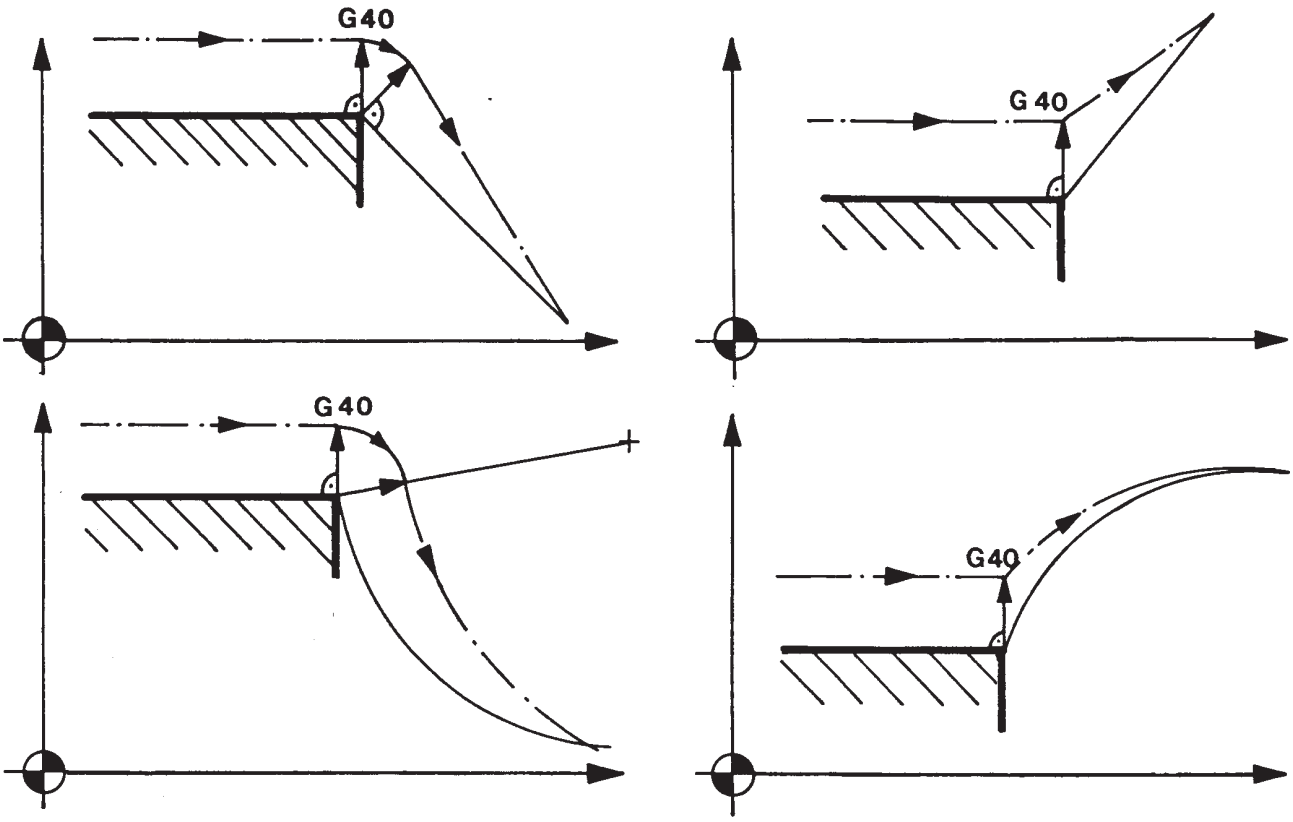


Figure 7-9

7.6 Switching off the correction

outer contour

inner contour



take off tangentially

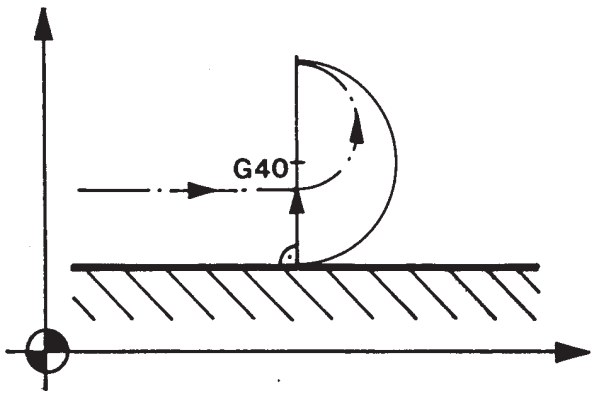


Figure 7-10

7.6 Switching off the correction (continued)

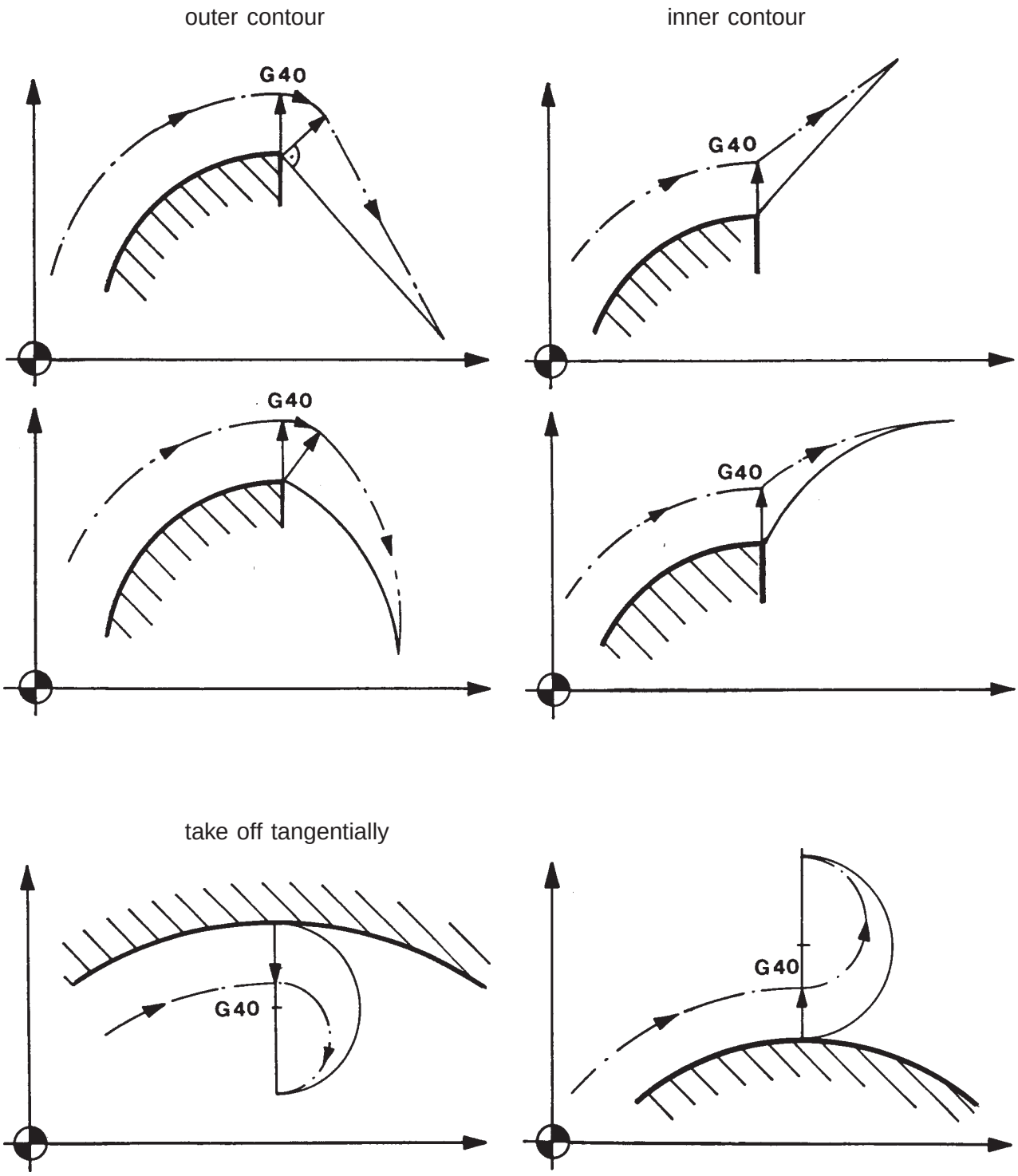


Figure 7-11

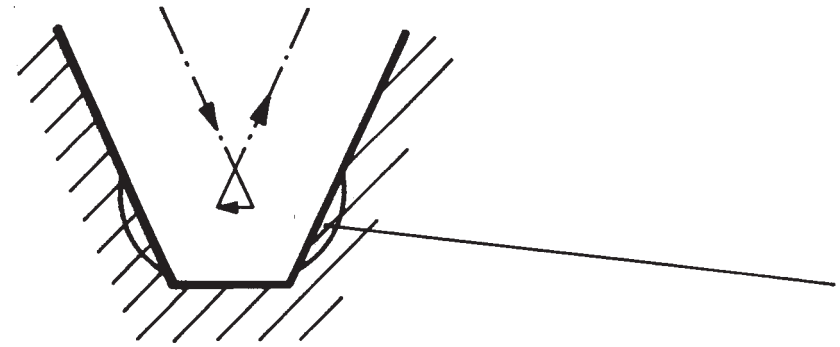
7.7 Special cases at inner contours

At inner contours undesirable contour errors can appear. In these cases the messages 1416 and 1420 are given out. The program is not interrupted.

The causes of these errors are:

1416 Too short travels in comparison with the dimension of the tool radius.
1420 Intersection not possible.

In the following figures the response of the control is illustrated for different cases.



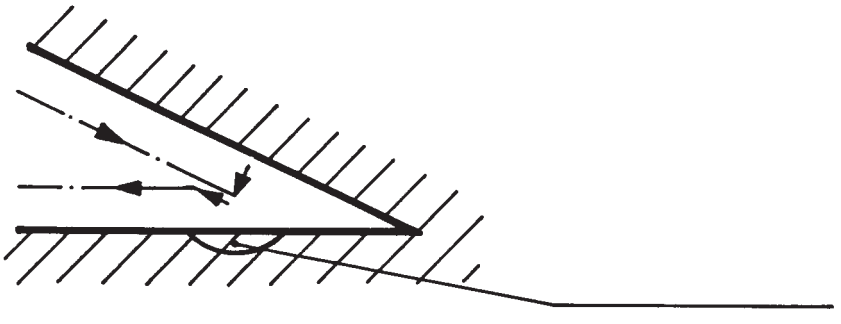
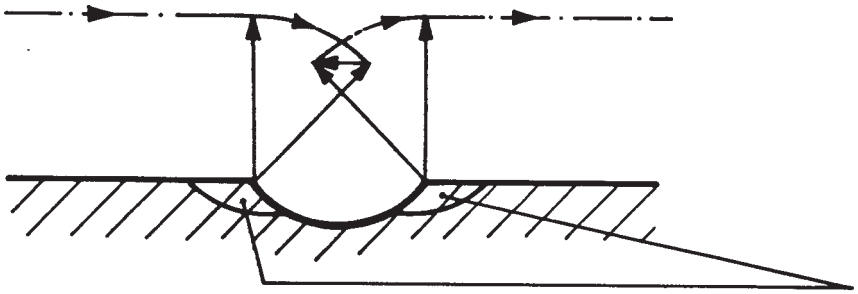


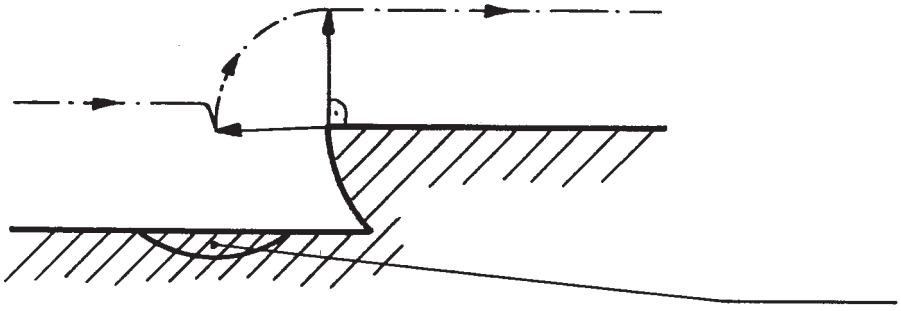
Figure 7-12

7.7 Special cases (continued)



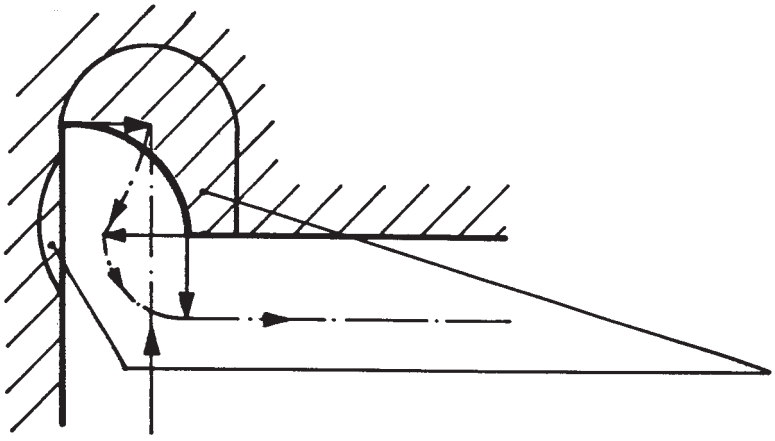
Message 1223

Contour error



Message 1223

Contour error



Message 1223

Contour error

Figure 7-13